

Design of turbomachinaries using numerical simulation results

E Gurina

Department of computational mathematics and computer modeling, Tomsk State University, Tomsk, 40 Lenin str., 634050, Russia

E-mail: elena.gyrina@gmail.com

Abstract. Computer simulation is the most modern method in the field of evaluating the parameters of a developed product in engineering problems. The essence of this method is to replace the real physical process that occurs during the operation of turbomachinery, with its mathematical model, and solve the problem with the help of modern computers. An approach with the replacement of a long (from several months to a year) and expensive development cycle by a numerical experiment (from several days to a month) will allow us to detect and eliminate the main flaws at the stage of product design, without realizing intermediate prototypes. Distributions of the main gas-dynamic parameters of the process studied (airflow velocity and pressure, mass flow rate) were obtained.

1. Introduction

Computer simulation is the most modern method in the field of evaluating the parameters of a developed product in engineering problems. The approach of replacing the high-cost development cycle with the numerical experiment allows revealing and eliminating the construction's main faults already at the stage of the product design. The paper presents the methodology for conducting the gas-dynamic calculation of an axial fan with the view to replace a full-scale experiment with a numerical one.

2. Geometric task description

The impeller machines such as fans serve for transforming the engine power into the gas (fluid) kinetic energy. The energy transformation occurs as a result of the fluid flowing around the blades [1]. The results of mathematical modelling allow revealing the main faults in the profile geometry of the blade's vane and evaluating aerodynamic performance of the whole construction already at the design stage.

The object under examination is an axial mine fan used for the ventilation of small minings (Figure 1). The fan's main elements are: cylindrical shell (CS) and rotor wheel (RW) where blades (B), flow straightener (FS), and electric driver (ED) are mounted. The fan's model is a fully-featured assembly comprising different parts, sub-assemblies, and standard elements [2].

The set task is solved with the following assumptions: the disturbances from the input grill at the fan inlet are disregarded, and all service openings, mounting hardware, as well as the shell's external elements are removed.

3. Physico-mathematical task description

The simulation is made for the process of three-dimensional stationary bumpy isothermal flow of incompressible air, which takes place in the flow section of the axial fan with the predetermined value



of the mass air flow rate. The task is to determine the difference in the total pressure developed by the fan. In the examined case the mathematical model describing the air flow in the fan flow section comprises the Reynolds-averaged continuity equation and Navier-Stokes equation [3]:

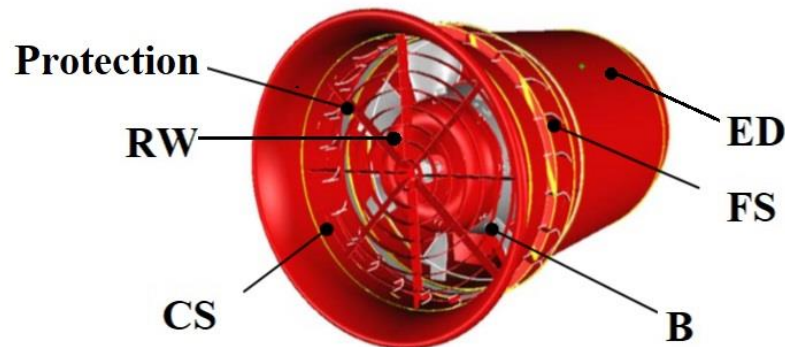


Figure 1. Simplified geometry of an axial single-stage fan:

CS – cylindrical shell, RW – rotor wheel, B – blades, FS – flow straightener, ED – electric driver.

$$\frac{\partial \bar{u}_j}{\partial x_j} = 0, \quad (1)$$

$$\frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu \frac{\partial \bar{u}_i}{\partial x_j} \right) - \frac{\partial}{\partial x_j} (\overline{u_i' u_j'}), \quad (2)$$

where \bar{u}_i – averaged projections of the velocity vector on the coordinate axis Ox_i , ρ – density, \bar{p} – pressure, ν – air kinematic viscosity, $u_i' u_j'$ – Reynolds stress tensor.

For the completion of simultaneous equations (1)-(2), two-parameter standard turbulence model $k - \varepsilon$ was used together with the gradient-diffuse Boussinesq hypothesis [4]:

$$-\overline{u_i' u_j'} = \nu_\tau \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij},$$

where δ_{ij} – Kronecker symbol. For the present task turbulence model $k - \varepsilon$ allows obtaining quality integral characteristic within a short period of time. The model comprises two transfer equations, one for the kinetic turbulent energy k and the other for its dissipation rate ε :

$$\frac{\partial \bar{u}_j k}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\nu + \frac{\nu_\tau}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P - \varepsilon, \quad \nu_\tau = C_\mu \frac{k^2}{\varepsilon},$$

$$\frac{\partial \bar{u}_j \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\nu + \frac{\nu_\tau}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} (C_{\varepsilon 1} P - C_{\varepsilon 2} E), \quad P = \nu_\tau \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \frac{\partial \bar{u}_i}{\partial x_j};$$

where P – generation of turbulent energy, ν_τ – turbulent viscosity. Coefficients of the turbulent model: $C_{\varepsilon 1} = 1.44$; $C_{\varepsilon 2} = 1.92$; $C_\mu = 0.09$; $\sigma_k = 1.0$; $\sigma_\varepsilon = 1.3$. Specified boundary and initial conditions are as follows: inlet full pressure $P_{\text{tot}} = 101\,325$; condition for adhesion on all stationary walls $u = 0$; mass air flow rate at the outlet $Q = 8.3\text{ kg/sec}$. The homogeneous distribution of $P_0 = 101\,325$ and $u = 0$ characteristics is taken as the initial approximation in the computational domain. The angular velocity of RW rotation is $\omega = -3000\text{ rev/min}$.

4. Construction of computational domain

Since the aim of the work is to replace a full-scale experiment, the fan geometry is regarded together with the geometry of a testing stand which is made as cylindrical branch tubes mounted prior to and after the fan, with the view of the flow stabilization. The geometry of inlet and outlet branch tubes is designed in accordance with the Russian State Standard 10921-90, <http://docs.cntd.ru/document/gost-10921-90>. It should be noted that it is only the periodic segment of the fan flow section that will be considered, because the fan model and, accordingly, the inlet/outlet branch tubes have a radial symmetry (Figure 3).

Figure 3 shows the three-dimensional periodic finite-volumetric computational mesh for the domain of the fan and the testing stand.

5. Approximation of differential problem

Numerical solution of simultaneous differential equations in partial derivatives is conducted on the basis of the finite volume method with the use of ANSYS Fluent software [5]. The finite-volumetric mesh contains 1 800 325 elements, with 30% of them accounting for the area around the RW blade (Figure 2). Figure 2 shows the surface mesh on the blade of the rotor wheel and on the periodic segment of the hub. The calculation elements are accumulated near the blade airfoil and all walls. The fields of velocity and pressure were first matched with the use of SIMPLE method, then by Coupled one. Approximation diagrams by space and time have the second-order accuracy.

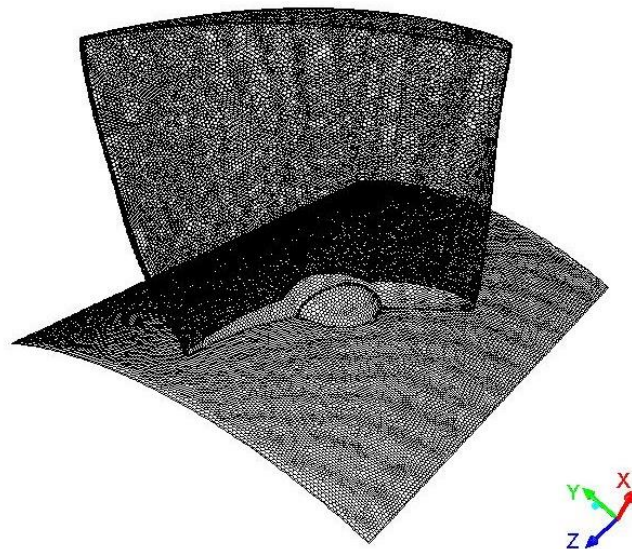


Figure 2. The finite-volumetric computational mesh in the area of the RW blade airfoil (surface mesh).

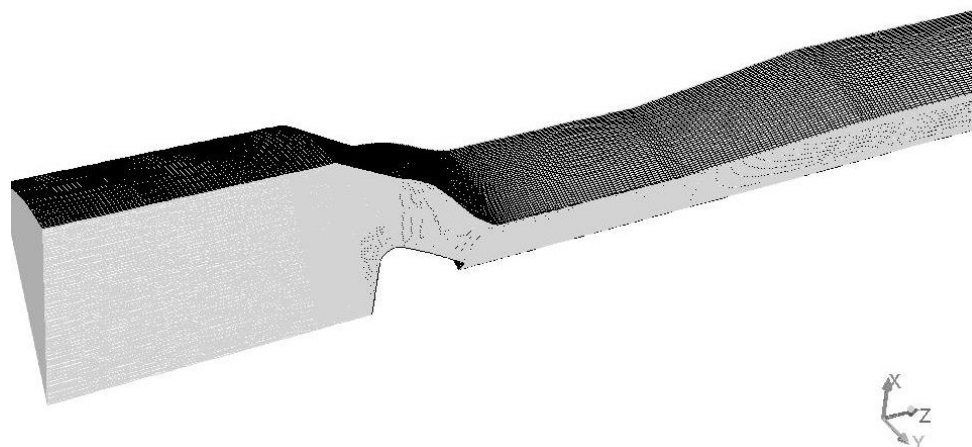


Figure 3. The finite-volumetric computational mesh for the domain (periodical segment).

6. Analysis of the obtained computational results

Based upon the known mass air flow rate and according to the constructed finite-volumetric mesh, the gas-dynamic calculation was conducted and the value of difference in the total pressure was obtained, which is 2 679 Pa. For the qualitative evaluation of the flow characteristics, the fields of velocities and pressures were constructed (Figures 4-6). Figure 4 shows the contours of axial velocity along the domain including blade of the rotor wheel. The contours of static pressure distribution on the blades of the rotor wheel and flow straightener are showed on the Figure 5. The field of velocity vectors in the horizontal section of the rotor blade is showed on the Figure 6. It should be noted that the flow in the blade-to-blade channel and around the blade profile is optimal, there are no vortexes or flow separations. The simulation results revealed the optimality of the blade's vane profile geometry and attached flotation around in terms of all main heights (foundation, midpoint, periphery).

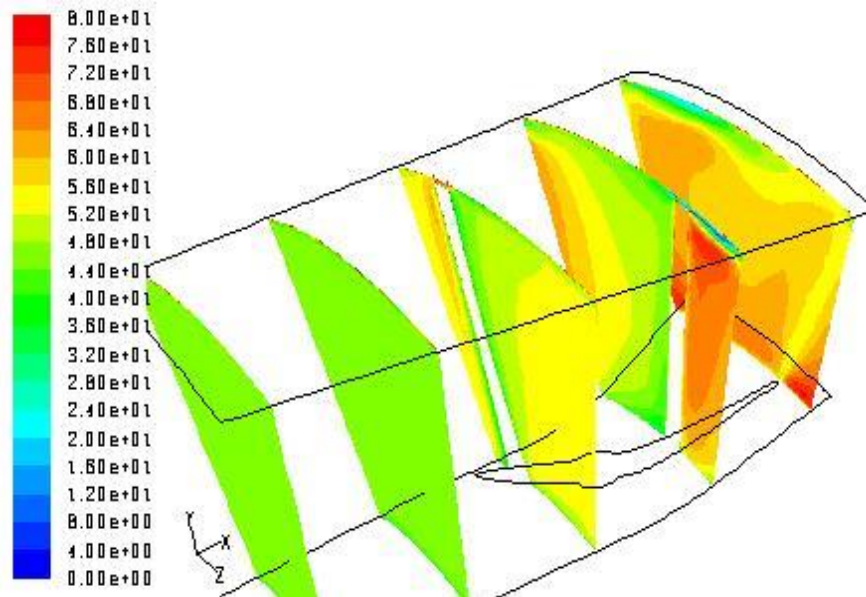


Figure 4. Contours of velocity, m/sec.

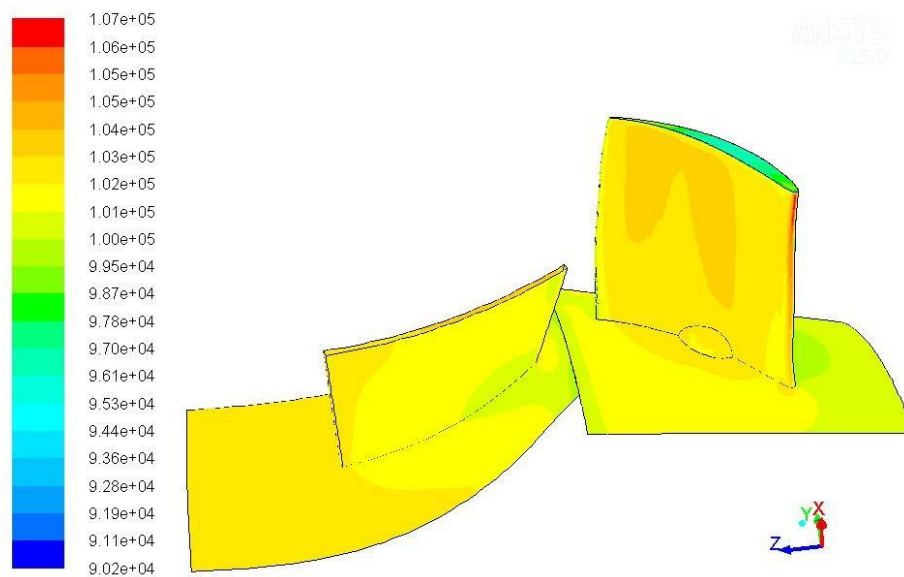


Figure 5. Distribution of static pressure.

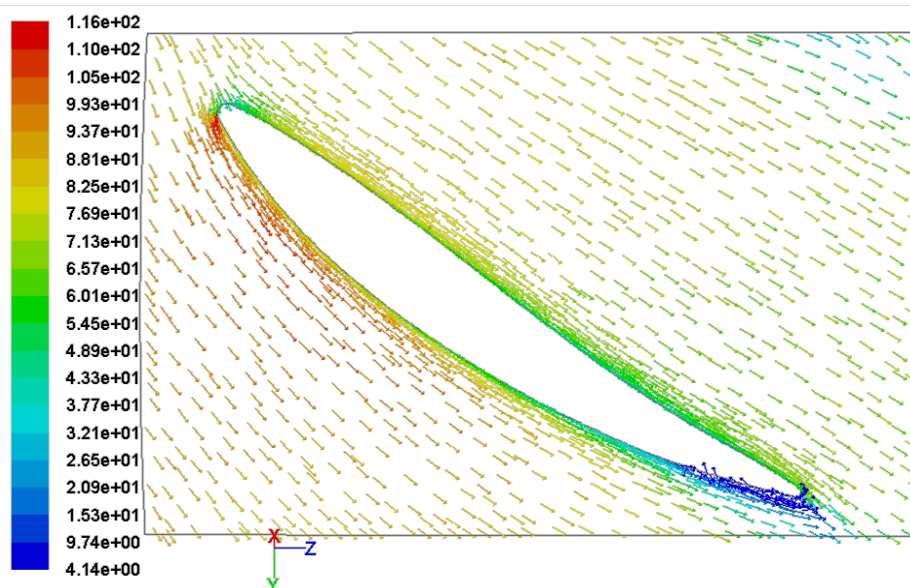


Figure 6. The field of velocity vectors (50% of the blade high), m/sec.

7. Conclusions

When conducting mathematical modeling, it is advisable to develop the calculation methodology by using existing fan models (to compare the calculated parameters with experiment data). After the methodology has been developed, it is possible to replace each prototype and natural experiment with a numerical experiment, and conduct virtual optimization until the fan parameters become optimal for this model. Summing up, let's say that mathematical modeling and the computer experiment associated with it are a worthy alternative to physical experiment, and are indispensable in cases where a physical experiment is impossible or difficult for some reason.

References

- [1] Castegnaro S 2018 *Designs* **2** (3) 20
- [2] McKenzie A B 1997 *Axial flow fans and compressors. Aerodynamic design and performance* Ashgate Publishing Limited: Aldershot, UK
- [3] Launder B E and Spalding D B 1974 *Computer Methods in Applied Mechanics and Engineering* **3** 269–289
- [4] Menter F R, Kuntz M and Langtry R 2003 *Turbulence, Heat and Mass Transfer* **4** 625-632
- [5] Frolov D 2015 *Engineering and technical journal ANSYS Advantage. Russian version* **11** 1-3

Reproduced with permission of copyright owner. Further reproduction prohibited without permission.